

THERMOHYDRAULIC EFFECT OF NOZZLE SIZE AND INTRODUCTION OF DIMPLED SURFACES IN HEAT SINK FOR THERMOELECTRIC REFRIGERATOR

MAHESH CHARAYA¹, S. SINGH² & RAJASEKAR. R³

¹PG Student, ME Department, Chandigarh University, Punjab, India

²Professor, ME Department, Chandigarh University, Punjab, India

³Professor, AE Department, Chandigarh University, Punjab, India

ABSTRACT

The computational study of water jet cooled hot side heat sink for thermoelectric refrigerator has been analyzed to study the effects of change in nozzle length (l) and diameter (d) on various parameters like the convective heat transfer coefficient, thermoelectric resistance, pressure drop and Nusselt number. It was observed from the analysis that nozzle configuration of l=3 mm and d=1 mm gave maximum value of the heat transfer coefficients as compared to other nozzle geometries considered at various flow rates. Computational studies have been done to analyze the heat sink performance by introducing dimple surfaces of three different diameters of 1mm, 2mm and 3mm at the inlet port of 8 mm. Results were further more analyzed for nozzle configuration of l=3 mm, d=1mm with respect to the temperature differences of target surface and exiting fluid, for various flow rates from 1-5 litres per minute. Results showed that heat sink model with dimpled surfaces of diameter 3 mm gives maximum Nusselt number and convective heat transfer coefficient; high Pressure drop at lesser thermal resistance.

KEYWORDS: Convective Heat Transfer, Dimple Surface, Heat Sink, Peltier Cooling & Thermoelectric

Received: Jun 11, 2017; **Accepted:** Jul 03, 2017; **Published:** Jul 07, 2017; **Paper Id.:** IJMPERDAUG20178

INTRODUCTION

Thermoelectric refrigeration is the latest growing technology based on Peltier element that is environment friendly and in which heat transfer can take place through solid and liquid medium [1]. Its development tends to eliminate the problem of leakage of refrigerant that is generally found for the case of vapour compression refrigeration system (VCRS) and also problem of bulky size like in case of vapour absorption refrigeration system (VARs). In Peltier element, there are a number of thermocouples arranged electrically in series and thermally in parallel through which when the electric current is passed, a temperature differential is generated across the two sides on the surface. The main advantage of using Peltier element is that it can be even installed in very small space of one cubic inch [1]. Thermoelectric refrigeration has a wide range of applications such as for medicine and electronics where precise temperature control is required, portable ice boxes, transport of perishable products, military, aerospace and many more. Thermoelectric refrigeration (TER) is not widely used due to its low coefficient of performance (COP around 0.38 – 0.45) in comparison to VARs (COP around 0.6 – 0.7) and VCRS (COP around 2.6 – 3.0) concluded by Riffat et al [2]. This is the reason it is not being considered for many of the practical applications. The COP of TERs can be enhanced if the efficient heat sink is used. Until now, many types of heat sinks have been developed like air cooled heat sink, free and submerged jet liquid cooled heat sink and micro jet heat sink. Manoj et al. [3] reviewed the development in TER and TEAC systems and found that these systems

can be used as a substitute to traditional refrigeration and air-conditioning systems, but due to less development of these systems, the efficiency of these systems is around 5 to 15 % in comparison to conventional systems, which have efficiency around 40 to 60 %. Wei et al [4] investigated that Capability of TEAC and TEHS driven by PV/T modules and found out that the COP of TEAC is more than 0.45. The COP of TERS and TEAC can be enhanced if the difference to the temperature between hot side and cold side of Peltier element is low. Cosnier et al. [5] presented a numerical model of TEAC and TEHS and also investigated it experimentally and found that COP between 1.5 and 2.0 can be achieved by maintaining the temperature difference of 5 °C and by applying an electric current of 4A. Shen et al [6] analyzed air-conditioning system based on thermoelectric radiant with an electric supply of 1.2 A and obtained COP of 1.77. The system can also be used as an air heating system just by reversing the input current. Zhou and Yu [7] found that in designing of TERS, an important role is played by the hot side heat sink and also the thermal conductance of the heat sink for hot side should be optimized. Hamed et al. [8] found that the cold side performance of the thermoelectric modules can be enhanced by adjusting the temperature of hot side by water flow at a constant rate, and then it can also be used as an air cooling system. Riffat et al. [9] found efficiency of thermoelectric refrigerators can be enhanced with an optimum design of the heat sink for hot side. It has also been found that the COP of TERS in which water is used to cool hot side heat sink doubles than TERS with the air cooled hot side heat sink. The water cooled heat sink can be of two types, one is the free jet impingement heat sink and other is the submerged jet impingement heat sink. Robinson and Schnitzler [10] found in experimental investigation that the submerged jet impingement heat sink gives the higher heat transfer coefficient than that of the heat sink with free jet impingement for a given pumping power. Robinson [11] concluded that due to higher pressure drop in case of the micro jet heat sink, the submerged jet impingement heat sink is better as for practical considerations. Single submerged jet impingement has been compared with multiple submerged jet impingement heat sink by many researchers. Tie et al. [12] stated that later gives more uniformity of temperature distribution on target surface and enhancement in heat transfer coefficient. Experimentally and numerically, it has been found that many factors like jet to target distance, impingement velocity, jet diameter, pressure drop, liquid flow rate effect the performance of the hot side heat sink.

Table 1

Nomenclature		s_w	Jet to Wall Distance (mm)
VCRS	Vapour compression refrigeration system	d	Nozzle diameter (mm)
VARS	Vapour absorption refrigeration system	l	Nozzle Length (mm)
TER	Thermoelectric refrigeration system	A_t	Exposed target area of heat source (mm^2)
TEHS	Thermoelectric heating system	Q	Heat transfer (W)
TEAC	Thermoelectric air conditioning	H	Convective heat transfer coefficient ($\text{W}/\text{m}^2\text{K}$)
COP	Coefficient of Performance	ΔT	Temperature difference between target surface and exiting fluid (K)
H	Jet to target distance (mm)	R_{th}	Thermoelectric Resistance (K/W)
s	Jet to jet distance (mm)	lpm	Litre per minute

In the present paper, convective heat transfer coefficient, Nusselt number, temperature variations were analyzed for various flow ranges. Modifications were made in the hot side heat sink based on Karwa et al. [13]. l and d has been varied to three different values of 1mm, 2mm and 3mm to analyze the effect on Convective heat transfer coefficient, thermoelectric resistance (R_{th}), Nusselt number and Pressure drop. Computational analysis were performed for the model with

introduction of dimpled surfaces at the inlet to the pipe for nozzle configuration of $l=3\text{mm}$, $d=1\text{mm}$ and comparative study were also made between the two model with/without dimpled surfaces.

COMPUTATIONAL MODEL

Multiple submerged jet impingement heat sink model was selected to study the performance of TERS. Deionized Water was used as cooling fluid. As shown in Figure 1(a), the water enters from inlet port and then reaches the plenum (conical section). At the base of plenum there are numbers of nozzles, arranged in 5 rows x 5 columns. Jet to jet centre distance (s) was given as 6.25mm and jet to outside wall distance (s_w) as 5.5mm. The walls in model were given thickness of 2mm. It was observed that chamfering and contouring at inlet of nozzle decrease the pressure drop in case of submerged heat sink. A fillet of 0.2 mm radius was given at the inlet of nozzles. Then water passes through nozzles and strikes on hot side of peltier element (which is the target surface). After striking the target surface with high velocity, the water exits perpendicular to the direction of target surface and then leaves from the model by outlet port. Interfacial thermal resistance was neglected as fluid directly strikes on target surface. There are two models, model (a) and (b), as shown in Figure 1. The model (a) is without dimples on which effect of change in nozzle length and nozzle diameter has been studied on heat transfer coefficient and other parameters. The model (b) is with dimpled surfaces. The effect of dimpled surfaces was studied on the temperature difference between target surface and exiting fluid by varying the inlet velocity. The various dimensions of the heat sink (in mm) are shown in Figure 1(b) and Figure 3. Length of the inlet/outlet port is 20mm, height of plenum is 15mm and the length of the connecting part from inlet port to plenum is also 20mm. The base of the heat sink model is $70 \times 70 \text{mm}^2$.

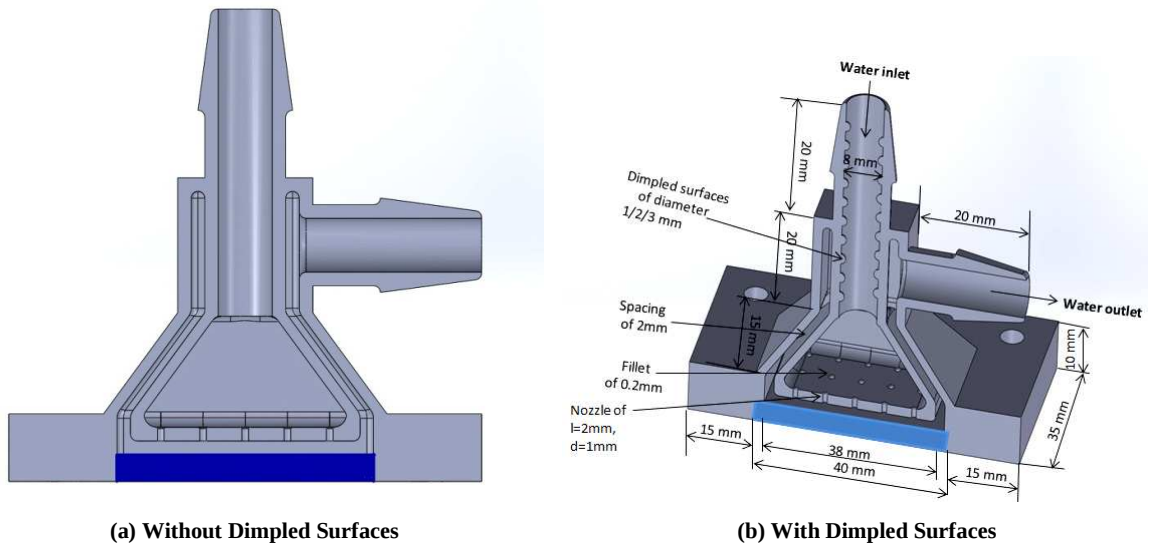


Figure 1: Sectional View of the Presented Heat Sink Model

CORRELATING EQUATIONS

Heat transfer coefficient was calculated by Fourier equation

$$Q = hA_t\Delta T \quad (1)$$

Where, Q denotes heat transfer rate, h denotes average convective heat transfer coefficient, ΔT denotes the difference of temperature between average temperature of target surface and bulk average temperature of the fluid, A_t denotes target surface area which is $38 \times 38 \text{ mm}^2$.

Nusselt number was calculated as

$$Nu = \quad (2)$$

Where, Nu denotes the Nusselt number, k denotes thermal conductivity of flowing fluid (water). l is the characteristic length which was assumed as the diameter of the nozzle.

Thermal resistance was calculated as

$$R_{th} = \frac{\Delta T}{Q} \quad (3)$$

Where, R_{th} denotes thermal resistance, ΔT denotes temperature difference between target surface and bulk average temperature of the fluid and Q denotes heat transfer rate.

BOUNDARY EQUATIONS

Volumetric flow rate was varied from 1 lpm to 5 lpm; accordingly, mass flow rate value was given as input. Outlet boundary condition was selected as environmental pressure. Constant heat flux rate of 34626 W/m^2 was given at target surface (50 W for $38 \times 38 \text{ mm}^2$) where the water coming from the jet strikes. Inlet temperature of fluid was set as 293.20 K . Model walls were assumed to be adiabatic. Conduction and radiation effects were neglected.

MODEL VALIDATION

Present model has been validated against the pressure drop readings over a range of Re (Reynolds number) from 847 to 4246 for nozzle of $d = 1 \text{ mm}$. It can be observed in Figure 4 that present computational model has the same trend of readings as observed in similar heat sink model proposed by Karwa et al [13]. The present work experienced lower pressure drop (7.6%) and some variation in trend as compared to Karwa et al. [13] due to introduction of fillet of radius 0.2 mm at the inlet of nozzle. The boundary conditions and nozzle size in both simulations are selected same.

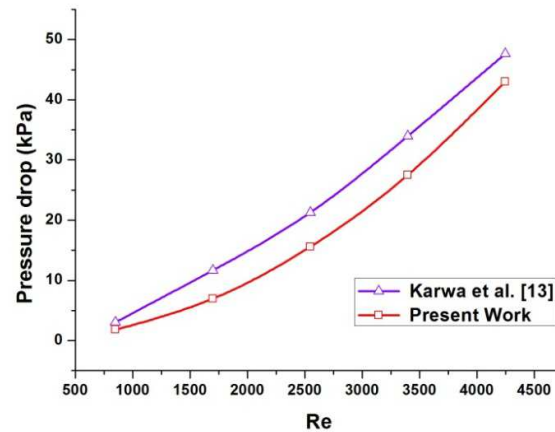


Figure 4: Model Validation by Comparing the Results of Present Computational Model with Karwa Et Al. [13] for Pressure Drop V/S Re

RESULTS AND DISCUSSIONS

Convective Heat Transfer Coefficient

The Figure 5(a) shows the variation in convective heat transfer coefficient (h) with the flow rate. It increases with increase in flow rate. The reason behind the increment of h can be considered due to mixing of striking fluid at fast rate. This decreases the temperature of target surface, but as the volumetric flow rate increases, the bulk average temperature of exiting fluid also decreases. In case of nozzle geometry $l=3\text{mm}$, $d=1\text{mm}$, which is the case of maximum value of convective heat transfer coefficient (in the model without dimple surface). In this case, as the flow increases from 1 lpm to 5 lpm, with 1 lpm increase in each time, value of h increases by 16.84 %, 6.62 %, 3.77 % and 3.16% respectively. Maximum increment in h with flow rate was observed when flow rate value changed from 1 lpm to 2 lpm

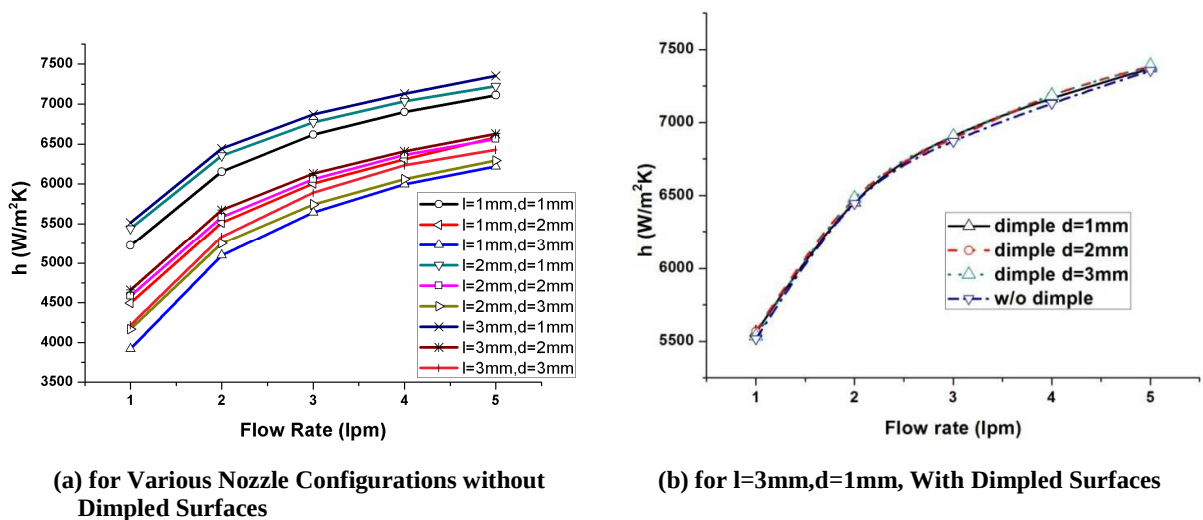


Figure 5: H V/S Flow Rate

It was also observed from the above Figure 5(a) that h increases with increment in length of nozzle (l) and with decrement in nozzle diameter (d) for a fixed flow rate. Increment of h w.r.t. length variation is lesser as compare to increment w.r.t. decrease in diameter. Maximum nozzle length assumed is 3mm and minimum nozzle diameter is 1 mm. there-

fore as we discussed, according to that maximum h value is observed in case of nozzle geometry $l=3\text{mm}$ and $d=1\text{mm}$. minimum value of h was observed at $l=1\text{mm}$ and $d=3\text{mm}$. the value of h increases by 40.77 % when nozzle size was changed from $l=1\text{mm}$, $d=3\text{mm}$ to $l=3\text{mm}$, $d=1\text{mm}$ at 1lpm. if decrease the diameter from 1mm to any lower value, there are chances of clogging of nozzles practically and nozzle length more than 3mm can also be restricted due to space constraint.

As shown in Figure 5(b), by introducing dimpled surfaces in the present model, at flow rate of 1 lpm h value decreases than that of without dimpled surface model, when dimple surface of 1mm, 2mm and 3mm introduced respectively. At flow rate of 2 lpm with dimpled surfaces of diameter 1mm, h value also decreases, but as dimpled surfaces of diameter 2mm and 3mm used h value increased at the same flow rate. When it was increased more than 2 lpm h value increases when dimple surface of 1mm, 2mm and 3mm used. Maximum increase in h was observed at flow rate of 4 lpm when dimpled surfaces of 3mm were used.

THERMAL RESISTANCE

As shown in Figure 6(a) it was observed that the thermal resistance (R_{th}) decreases with the increase in flow rate. Decrement in R_{th} follows the trend of polynomial of 2nd order (with the regression coefficient around 0.95-0.98). The reason for the decrease of R_{th} with the increase of flow rate is attributed due to more fluid component striking the target surface thus leading to less temperature difference ΔT (using $R_{th} = \Delta T/Q$). At nozzle geometry $l=1\text{mm}$, $d=3\text{mm}$ when flow rate was varied from 1 lpm to 5 lpm, the decrement of R_{th} was observed to be 21.60- 3.58 %.

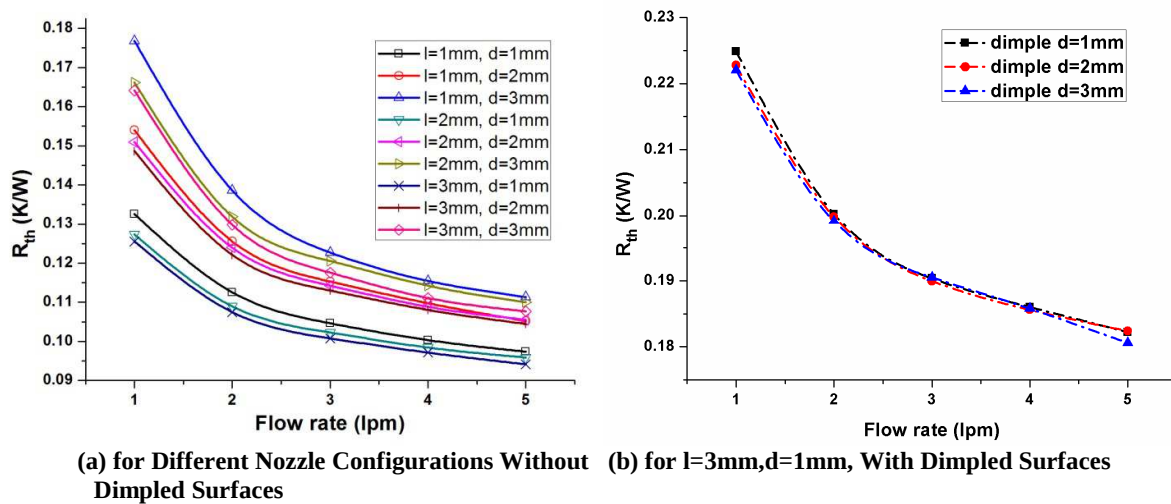


Figure 6: Thermal Resistance V/S Flow Rate

It was observed that with increase in flow rate from 1 lpm to 5lpm, at constant length of 1mm, when diameter of nozzle increase from 1 mm to 3mm, the thermal resistance increases by 14.32–33.33% and at constant diameter of 1 mm, when length of the nozzle increase from 1mm to 3mm, it decreases by 3.32–5.29%. At constant heat flux and length of nozzle as the diameter increases R_{th} increases, this may be attributed due to less striking velocity of the fluid on the target surface thereby decreasing the convective coefficient which leads to increase in ΔT . The increase in R_{th} with an increase in d is much more than that of decrease in l . Minimum thermal resistance is observed at nozzle geometry $l=3\text{mm}$, $d=1\text{mm}$ at flow rate of 5 lpm and maximum thermal resistance at $l=1\text{mm}$, $d=3\text{mm}$ with flow rate of 1 lpm. R_{th} decreases by 15.43% when nozzle size is changed from $l=1\text{mm}$, $d=3\text{mm}$ to $l=3\text{mm}$, $d=1\text{mm}$ at flow rate of 5lpm.

As shown in Figure 6(b) when dimpled surfaces were introduced in the heat sink model, R_{th} was increased. With

increase in diameter of dimpled surfaces up to 3mm, R_{th} decreases, but not much decrement was observed. Diameter of dimpled surfaces can be increased more than 3mm, to observe the variation in R_{th} .

PRESSURE DROP

It was observed from the Figure 7(a) that as flow rate increases, the Pressure drop (dP) also increase and it follows the polynomial trend (of 2nd order). At a given flow rate and fixed nozzle length, dP was observed to increase with decrease in diameter of the nozzle. This increment in dP with decrease in d at a given flow rate can be attributed due to lesser the diameter at the nozzle, more will be the obstruction in the flowing fluid. It was also observed to increase with increase in nozzle length at fixed flow rate and nozzle diameter. The increment in dP v/s flow rate with an increase of l is observed to be at a higher rate when $d=1mm$. Maximum pressure drop is observed at nozzle geometry of $l=3mm$, $d=1mm$ with flow rate of 5 lpm. Minimum pressure drop was observed at nozzle geometry of $l=1mm$, $d=3mm$ with flow rate of 1 lpm. At this nozzle geometry, minimum pressure observed was 0.29 kPa when flow rate was 1lpm and maximum pressure drop observed was 6.73 kPa when flow rate was 5 lpm. When nozzle geometry was changed from $l=1mm$, $d=3mm$ to $l=3mm$, $d=1mm$ at 5lpm, dP increased to 46.92 kPa. At constant flow rate, nozzle length and diameter ($l=3mm$ and $d=1mm$), with the introduction of dimpled surfaces, dP was observed to be increased by 0.5-2.5% when dimples diameter changed from 1 mm to 3mm. Maximum pressure drop of 47.37kPa was observed with dimple surface of 3mm at flow rate of 5 lpm.

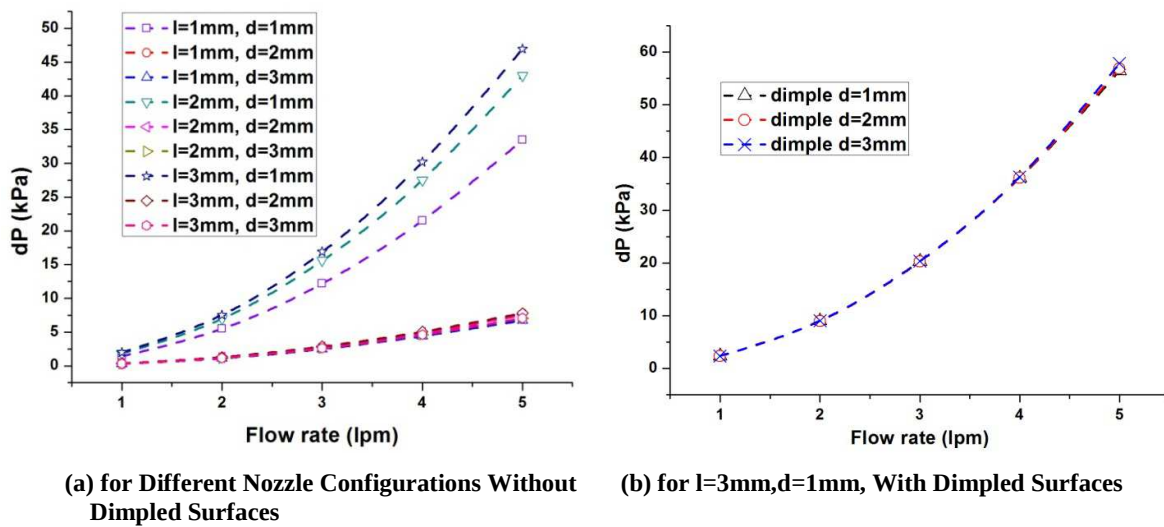
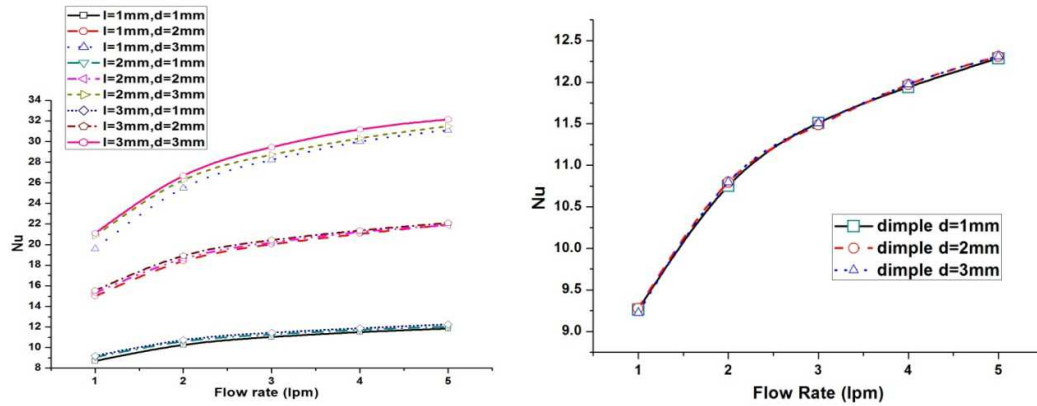


Figure 7: Dp V/S Flow Rate

NUSSELT NUMBER

Nusselt number (Nu) tells us about the enhancement in heat transfer rate when we give motion to stagnate fluid. As shown in Figure 8(a) Nu was observed to increase with the increase in flow rate. Its variation followed the logarithmic trend (with the regression coefficients around 0.99). It was also observed that Nu was increased with the increase in l at constant flow rate and d . Increase in Nu with increase in d was also observed at constant flow rate and l . At a constant $l=3mm$ and flow rate of 1lpm, when the nozzle diameter was changed from 1-3mm, the percentage increase in Nu was observed as 129%. At a constant nozzle diameter of 1mm and flow rate of 1lpm when the nozzle length was changed from 1-3mm, the percentage increase in Nu was observed as 5.58%. Thereby it was analyzed that Nu increases more rapidly with the nozzle diameter than that with nozzle length. Maximum Nu was observed at nozzle geometry of $l=3mm$, $d=3mm$.

As shown in Figure 8(b), by introducing the dimpled surfaces in the heat sink model, not much variation was observed in Nu. When flow rate was more than 2lpm, it was observed that Nu increases with increase in diameter of dimpled surfaces. Maximum value of Nu was found for dimpled surfaces of diameter 3mm.



(a) for Different Nozzle Configurations Without Dimpled Surfaces (b) for l=3mm,d=1mm, With Dimpled Surfaces

Figure 8: Nu V/S Flow Rate with Change in Nozzle Geometry (Without Dimpled Surfaces)

TEMPERATURE DIFFERENCE BETWEEN TARGET SURFACE AND EXITING FLUID

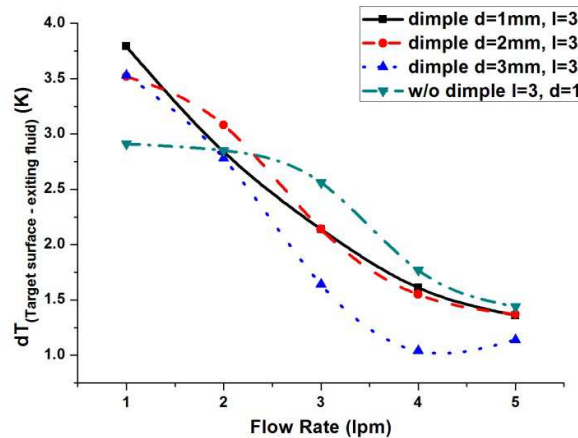


Figure 9: $dT_{\text{Target Surface - Fluid}}$ v/s Flow Rate with Change in Nozzle Geometry

Temperature difference between target surface and exiting fluid depends on various parameters, including but not limited to flow rate of cooling liquid, inlet fluid temperature and nozzle geometry. It was analyzed in the model without dimpled surfaces that nozzle geometry of l=3mm and d=1mm gave lower thermal resistance, and higher heat transfer coefficient. At this nozzle geometry, the pressure drop was also high, which can be reduced if the lesser flow rate may be used. Dimpled surfaces were introduced at the inlet section of the model for this nozzle geometry. The results of four configurations of the model were observed which are shown in Figure 9. The first configuration was the heat sink model without dimpled surface; in that situation, the temperature difference between the target surface and exiting fluid was lowest at flow rate of 1 lpm. The second configuration was the model with dimpled surfaces of diameter 1mm, in which the temperature difference was observed to be highest at same flow rate of 1 lpm. The third configuration was the model with dimpled surfaces of diameter 2mm, in which the temperature difference observed was closer to the model with dimpled surface of

diameter $d=3\text{mm}$ (which is the fourth configuration of the model) at flow rate of 1 lpm. With the further increment of flow rate to 2 lpm and more, the temperature difference was observed to be lesser in case of the model with dimpled surfaces of 3mm diameter. Minimum temperature difference was observed for this geometry at flow rate of 4 lpm.

CONCLUSIONS

The simulation study of water jet cooled submerged jet hot side heat sink has been done to analyze the effect of change in nozzle length and diameter on various parameters (Convective heat transfer coefficient, Thermal resistance, pressure drop, Nusselt number). With the study of these parameters, it has been observed that

- Maximum convective heat transfer coefficient was observed for the model with dimpled surfaces of diameter 3mm at flow rate of 5 lpm.
- Minimum thermal resistance was observed in the same conditions for maximum heat transfer coefficient.
- Minimum pressure drop was found in the model without dimpled surfaces at nozzle geometry of $l=1\text{mm}$, $d=3\text{mm}$ with flow rate of 1 lpm.
- Maximum Nusselt number was observed for model without dimpled surfaces at nozzle geometry of $l=3\text{mm}$, $d=3\text{mm}$ with flow rate of 5 lpm.
- Minimum temperature difference between target surface and exiting fluid was observed for the model with dimpled surfaces of diameter 3mm at flow rate of 4 lpm.

In this heat sink model, jet pattern (square array) can be modified to get the more uniform temperature on target surface, dimpled surfaced of higher diameter (than 3mm) can also be designed to study the effect on heat sink model parameters and turbulence effect may be studied. Thermoelectric heat sink model's plenum design can be modified to enhance the model's performance.

REFERENCES

1. Melcor TE handbook, https://home.zhaw.ch/~fusa/PSS_VLE_C/CHAPTER_03/CASES/Materials/Melcor_TE_Handbook.pdf
2. Riffat S.B. and Guoquan Qiu "Comparative investigation of thermoelectric air-conditioners versus vapour compression and absorption air-conditioners", *Applied Thermal Energy*, Vol.24, No.14-15, (2004), pp:1979-1993.
3. Rawat M.K., Chattopadhyay H., Neogi S., "A review on developments of thermoelectric refrigeration and air conditioning systems: a novel potential green refrigeration and air conditioning technology", *IJETAE*, Vol.3, No.3 (2013), pp:362-367.
4. He W., Zhou J., Hou J., Chen C. and Ji J., "Theoretical and experimental investigation on a thermoelectric cooling and heating system driven by solar". *Applied Energy*, vol.107, (2013), pp:89-97.
5. Cosnier M., Fraisse G., Luo L., "An experimental and numerical study of a thermoelectric air-cooling and air-heating system". *International Journal of refrigeration*, Vol.31, (2008), pp:1051 – 1062.
6. Shen L., Xiao F., Chen H. & Wang S., "Investigation of a novel thermoelectric radiant air-conditioning system", *Energy and Buildings*, Vol.59, (2012), pp:123-132.
7. Zhou Y. and Yu J., "Design optimization of thermoelectric cooling systems for applications in electronic devices", *Inter. Jour. of Refrigeration*, Vol.35, (2012), pp: 1139-1144.
8. Dizaji H.S., Jafarmadar S., Khalilarya S., Moosavi A., "An exhaustive experimental study of novel air-water based thermoelectric heat sink", *Applied Thermal Engineering*, Vol.107, (2016), pp:1051-1062.

- lectric cooling unit”, *App. Energ.*, Vol.181, (2016), pp:357-366.
9. Riffat S.B. and Qiu G.Q., “Design and characterization of a cylindrical, water-cooled heat sink for thermoelectric air-conditioners”, *Intern. Jour. of Energ. Research*, Vol.30, No.2, (2006), pp: 67-80.
 10. Robinson A.J. and Schnitzler E., “An experimental investigation of free and submerged miniature liquid jet array impingement heat transfer”, *Experimental Thermal and Fluid Science*, Vol.32, (2007), pp:1-13.
 11. Robinson, “A Thermal-Hydraulic Comparison of Liquid Microchannel and Impinging Liquid Jet Array Heat Sinks for High-Power Electronics Cooling” *IEEE Transac. on Components and Packag. Technolog.*, Vol. 32, No. 2, (2009), p: 347-357
 12. Tie P., Li Q., Xuan Y., “Investigation on the submerged liquid jet arrays impingement cooling” *App. Therm. Eng.*, Vol. 31, (2011), p: 2757-2763.
 13. Karwa N., Stanley C., Intwala H., Rosengarten G., “Development of low thermal resistance water jet cooled heat sink for thermoelectric refrigerators”, *App. Therm. Eng.*, (2016), doi: <https://dx.doi.org/10.1016/j.applthermaleng.2016.06.118>

